Open Access

SCHOLARENA

Journal of Mathematical and Statistical Analysis

CASE REPORT

Ultimate Strength Calculation and Destructive Modes Extraction in a Box Girder Model (with ANSYS)

Jahangiri S*

Australian Maritime College, University of Tasmania, Australia

***Corresponding author:** Jahangiri S, Australian Maritime College, University of Tasmania, Australia, Tel: +0061 4 88 574 685, E-mail: sanaz.jahangiri@utas.edu.au

Citation: Jahangiri S (2019) Ultimate Strength Calculation and Destructive Modes Extraction in a Box Girder Model (with ANSYS). J Math Stat Anal 2: 203

Abstract

Of the most important factors in designing ships, is to analyze the structural behavior and predicting the exact ultimate strength of them. There are many researches done in this field. One of the methods is to simulate the ship's behavior with a similar box girder (under loading) until the ultimate strength. In my research, I simulated a box girder by finite element software (ANSYS). After applying boundary conditions, and a proper loading, I did the ultimate strength gradual analysis and extracted the destructive modes. Finally, I compared my results to other researches, and the conclusions showed that the ultimate strength I gained by this method was compatible to the amount gained by experimental tests on this model.

Keywords: Ship Design; Ultimate Strength; ANSYS

Background

Analyzing the structural behavior of ships and predicting their exact ultimate strength is one of the most important factors in ship design [1]. In this field, numerous research studies have been done. One of the methods is to simulate the behavior of the ship up to the ultimate strength with a similar box girder (under loading). In my research, I used finite element software (ANSYS) to simulate a box girder. I did the ultimate strength gradual analysis and extracted the destructive modes after applying boundary conditions and proper loading. Finally, I compared my results with other studies, and their findings showed that the ultimate strength I achieved from this method was compatible with the amount gained from this model's experimental tests.

I should explain Akhra's experimental test, his assumption, model dimension, material, initial residual stress, initial geometric deformations, and his loading conditions firstly and briefly in order to explain my whole job in the personal Workplace Activity part [2].

A ship's hull is often subjected to forces due to waves. Due to the hugging phenomenon, a ship's upper deck and bottom are under tension and pressure, respectively, whereas, in the sagging phenomenon it is vice versa. In the simulation method, the ship's behavior is assumed to be comparable to a simple beam. Consequently, the longitudinal strength of a ship is similar to the longitudinal strength of a beam. To do this, a beam's area must be selected according to the ship's area specifications. Akhra's used the box girder to simplify the hull's area, as is presented in Figure 1.

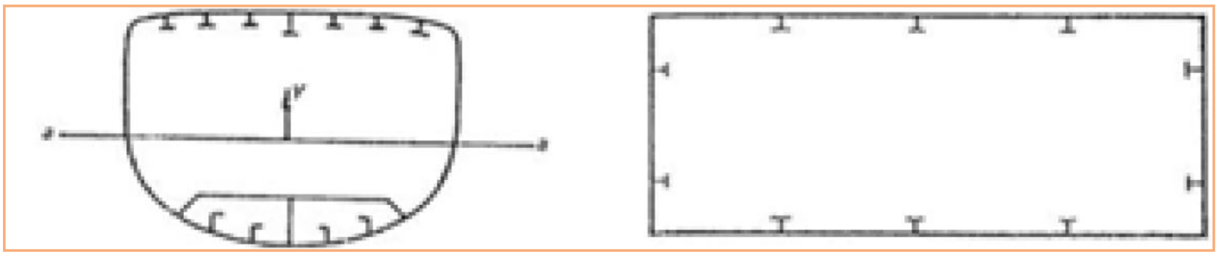


Figure 1: The box girder to simplify the hull's area

Akhra's built a box girder as explained above, which represents a structural order of a typical ship, and he did a test on it to gain the ultimate strength of the box girder. These kinds of tests were done until it was no longer possible to witness the ship's structural behavior. As presented in Figure 2, the box girder consists of three sections, namely, the test, buffer, and outer.

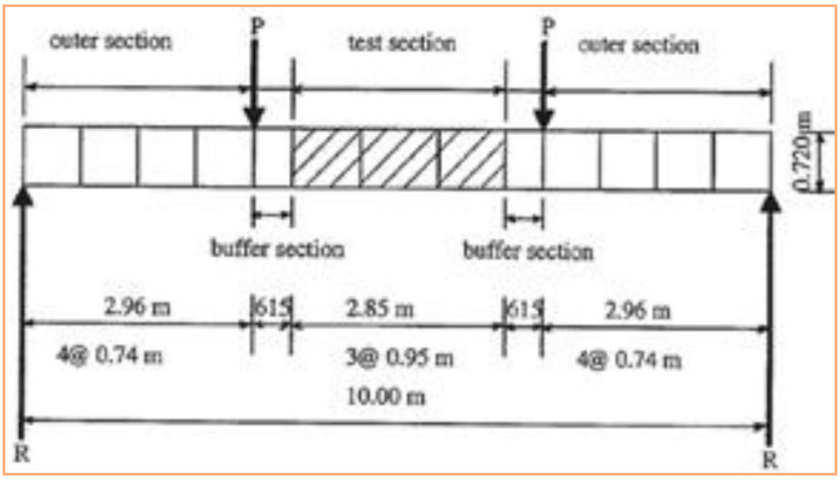


Figure 2: The three sections of the box girder

The test section is under immense pressure bending up to failure. The box girder cross-section and its details are presented in Figure 3.

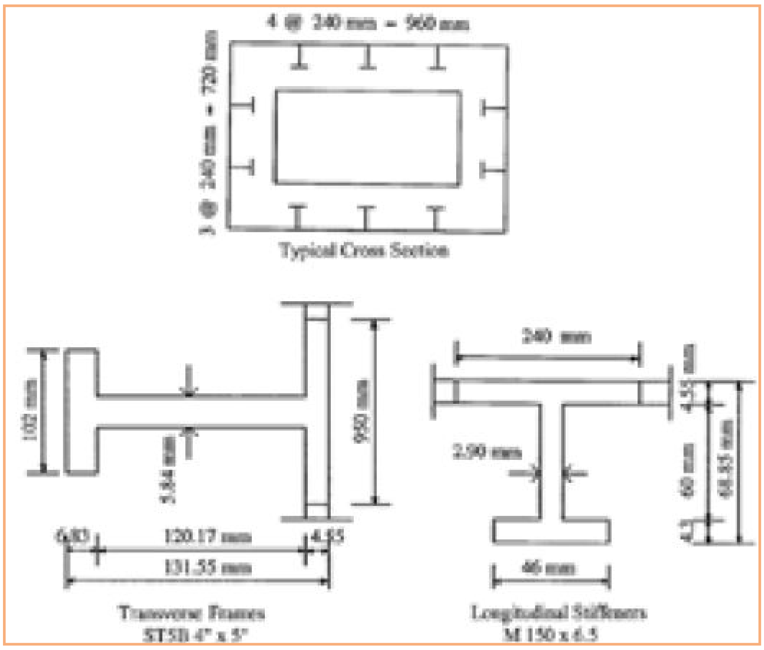


Figure 3: The box girder cross-section and its details

The model was designed in such a way that it prevents any failure mode prior to the local buckling of plates and stiffeners. In this built box girder model, interframe buckling failure under longitudinal bending moment is the most important failure mode for the typical ship with longitudinal framing. By gradually increasing the load, the central plate of the deck buckles among the stiffeners. By increasing the loading, even more, the central plate transfers the extra load to the nearby longitudinal stiffeners. Consequently, these stiffeners will buckle plate strips attached to them. Different plate strips buckling transfer the load to box corners. It is supposed that this inelastic instability happens, preceding the tensioned box girder's section reach to the plastic mode. The Akhra's box girder is 10 m long, and 960 mm × 720 mm cross-sectionally. The plate thickness in the test section is 4.55 mm, and 9.53 mm in the other sections. 4 different types of mild steel were used in the modeling. Considering the tensile tests, their details are given in Table 1.

Type of steel and material specifications	Nominal yield stress (MPa)	Average yield stress (MPa)	Average ultimate stress (MPa)	Avenge Young's modulus of elasticity (GPa)
Buffer—outer section plating; CSA G40.1, grade 38WT Cat5	250	365	498	204
Longitudinal stiffeners; ASTM A-36	250	318	466	205
Transverse frames; ASTA1 A-588 Grad B	350	417	531	199
Test section plating; SAE 1010	250	294	404	212

Table 1: Tensile test specifications

According to cross sections of longitudinal stiffeners, the rise in heat due to welding the webs to the plates is so significant that it causes great strain residual and consequently, considerable residual stress which plays a critical part in the box girder performance. As a result, Akhras also took this into consideration. The measurements of initial deformations due to welding the box girder are presented in Figure 4.

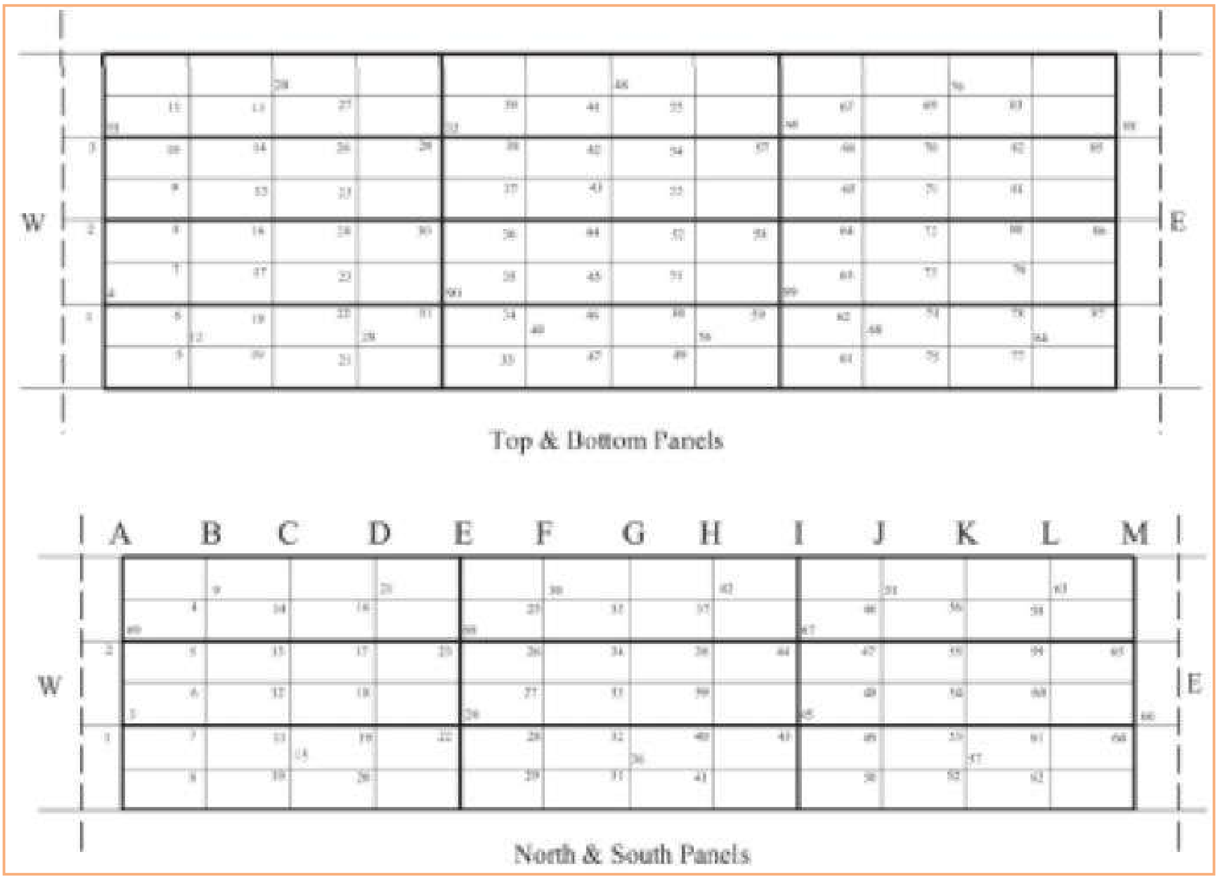


Figure 4: Measurements of initial deformations

Moreover, the initial deformation in the middle of the test section (where the buckling occurs) is presented in Figure 5.

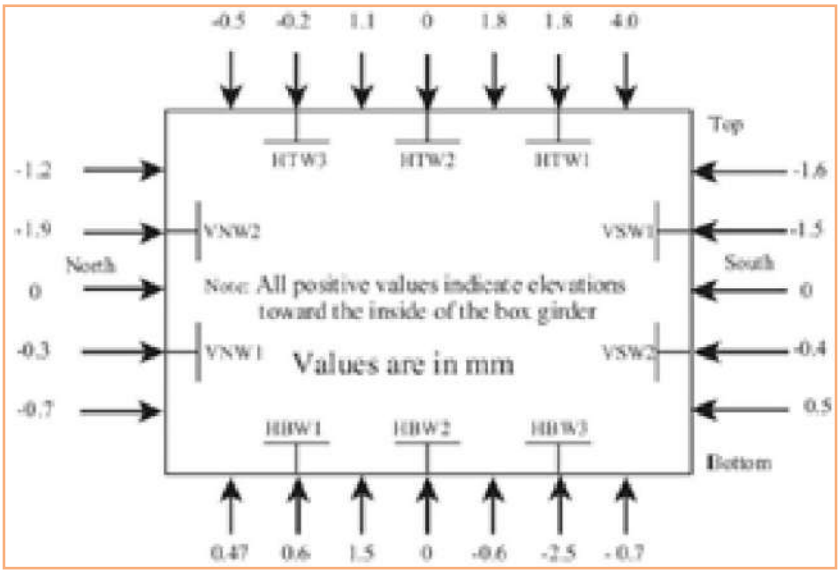


Figure 5: Initial deformation in the middle of the test section

Akhra's needed to create a pure bending so as to create the buckling in the upper plate before any kinds of failure happen. To do this, transferring the load from where it is applied to the test section, must be done uniformly. Thus, the middle section is considered 615 mm, and two big outriggers are placed exactly under the loading beams. Moreover, to ensure a pure bending, Akhras used wedge-shaped prisms and a roller as supports. Akhra's used three methods to calculate the ultimate strength of the box girder. These methods were: the theory method (following the Faulkner formula), the FABSTRAN software, and experimental testing. The ultimate moment gained from the theory method (without considering deformations and initial residual stresses) was 1250 kNm. This moment from the FABSTRAN software (considering deformations and initial residual stresses) was 1300 kNm, and finally, the moment from the experimental testing (under the applied loading of 836 kN) was 1238 kNm. In this paper, a box girder was simulated by finite element software (ANSYS). After applying boundary conditions and a proper loading, the ultimate strength gradual analysis was done and the destructive modes were extracted. Finally, the results were compared to those of other research studies. The findings obtained have numerous applications in engineering in terms of investigating the structural behavior of the exact ultimate strength.

Methodology

I simulated the box girder in my research using the finite element method. In my model, I considered Akhra's model-like dimensions, thicknesses, cross-sections, and material. I analyzed the model's ultimate strength under the bending loading after applying the proper boundary condition. Finally, I compared the results of my simulation with those of other researchers.

I used the ANSYS software to simulate and analyze my model [3].

I also used the SHELL181 element in all sections of my model. This element has four nodes and 6 degrees of freedom in every node. This element is suitable for linear analysis, large rotating analysis, and nonlinear analysis with a large strain.

I simulated only the left half of the Akhras box girder, which means that my analyzed model length is 5 m. My completed simulated model is presented in Figure 6.

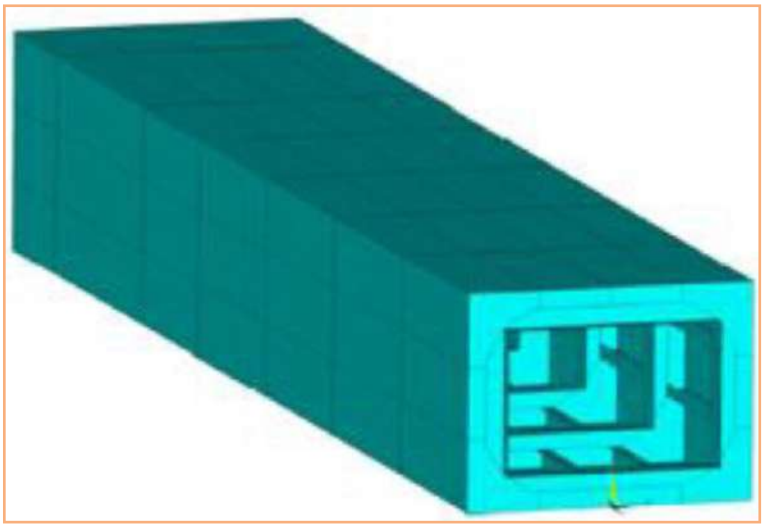


Figure 6: Completed simulated model

In Figure 7, a longitudinal view from the inside of the model is shown.

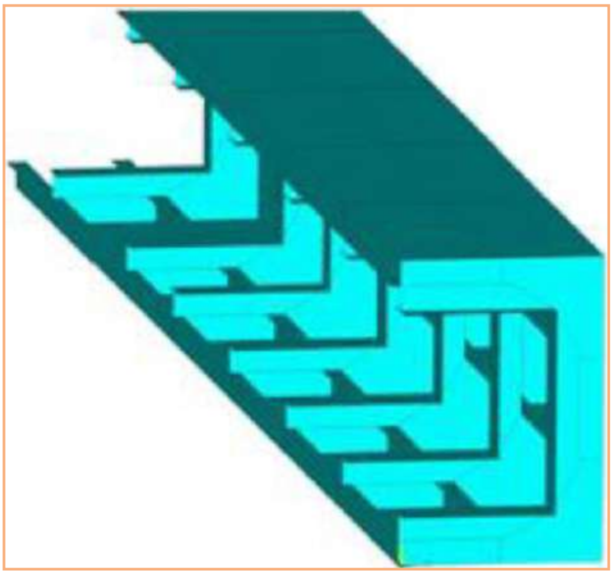


Figure 7: The longitudinal view from the inside of the model

I modeled longitudinal stiffeners, and the transversal frames are presented in Figure 8.

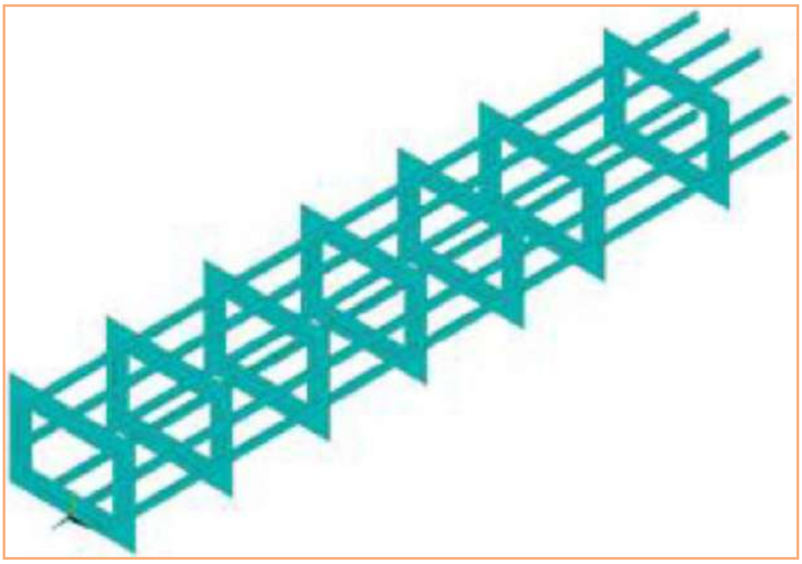


Figure 8: Transversal frames

The material I used in my model was like the Akhras one (which I showed before in a Table). I chose the material in the software as bilinear, and as a sample, the stress versus strain graph for the material I used in the test section plate is presented in Figure 9.

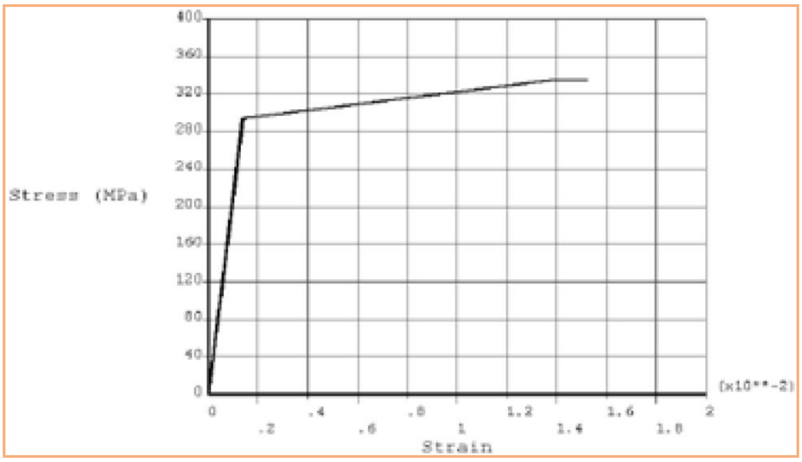


Figure 9: The stress versus strain graph

I used various materials in different places that are shown in Figure 10.

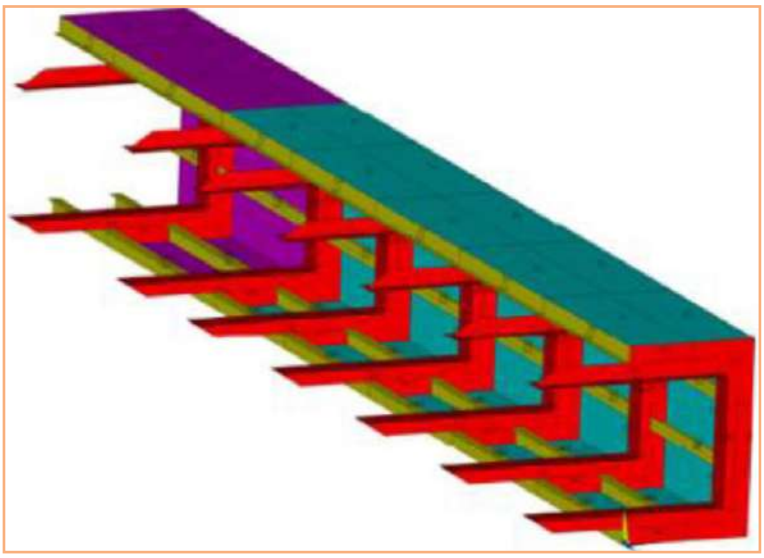


Figure 10: Materials applied in different sections

To give a detailed view of the above figure, Table 2 presents the various materials that I used in my finite element model.

Position Used in	Average Yield Stress (Mpa)	Average Young's Modulus of Elasticity (Gpa)	Color Display
Buffer-Outer Section Plating	365	204	
Longitudinal Stiffeners	318	205	
Transverse Frames	417	199	
Test Section Plating	294	212	

Table 2: Models specifications

As shown in Figure 3 and as explained in the Akhras model dimensions formerly, six different thicknesses were used in the Akhra's model, which I applied them in the model too (Figure 11).

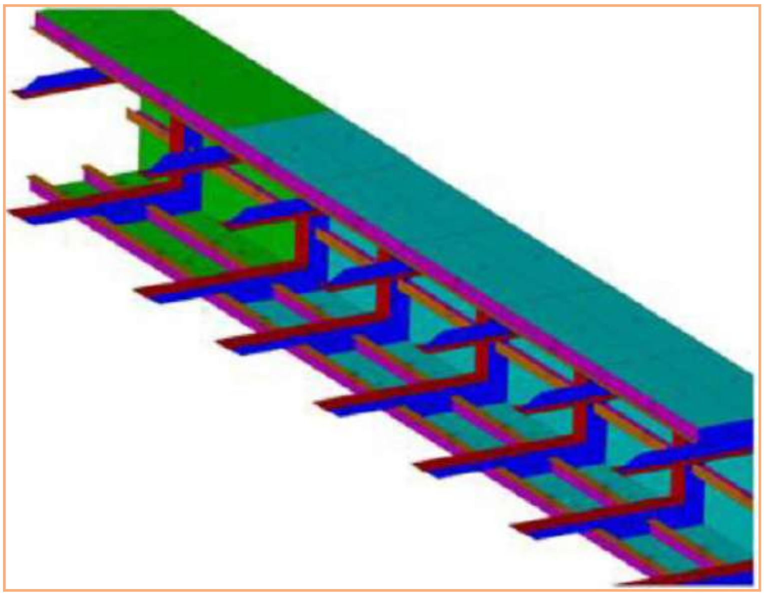


Figure 11: Different thicknesses applied in the modeling

To give an in-depth view of the above figure, various thicknesses that I used in my finite element model are shown in Table 3.

Thickness Number	Thickness (mm)	Position Used in	Color Display
1	9.53	Buffer-Outer Section Plating	
2	5.84	Transvers Frame's Web	
3	6.83	Transvers Frame's Flange	
4	4.3	Longitudinal Stiffener's Flange	
5	2.9	Longitudinal Stiffener's Web	
6	4.55	Test Section Plating	

Table 3: Different thickness specifications

Results

In this paper, the finite element software (ANSYS) simulated a box girder. The ultimate strength gradual analysis was performed after applying boundary conditions and proper loading, and afterward, the destructive modes were extracted. The results were finally compared with those of other studies. The results obtained have numerous engineering applications in investigating the structural behavior of the model.

In the nonlinear analysis of my finite element model, I began by applying the initial deformation to my model (as explained in the fifth Figure). Then, regarding the simulation of the left half of the Akhra's model, I constrained the left end of my model in the width and height directions, and in the right end, I considered symmetry boundary condition. After I did the static analysis, initial deformation happened in the buckling cross-section of the model, which can be seen in Figure 12.

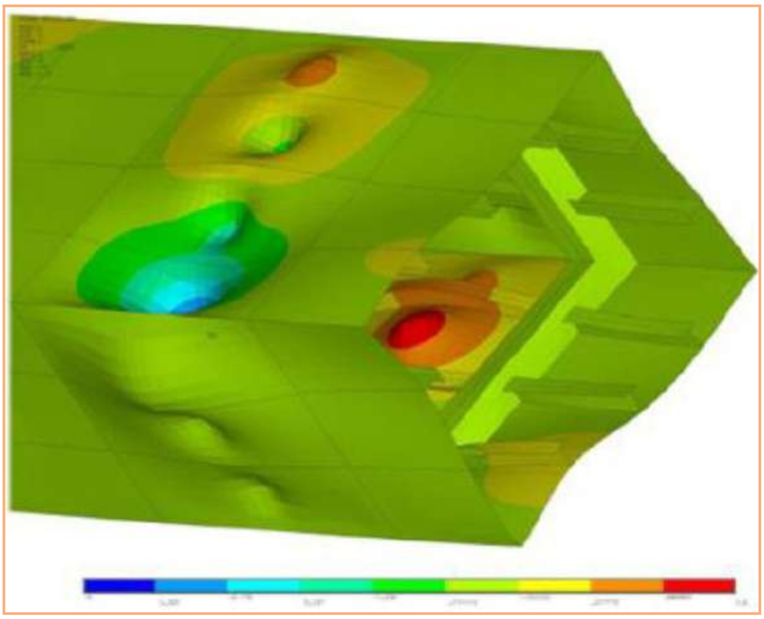


Figure 12: **I**nitial deformations in the buckling cross-section of the model In my simulation, I did not consider the initial residual stress.

As I said before, I used the SHELL181 element in my model. The element model is presented in Figure 13, and as it got closer to the test section, I used smaller elements.

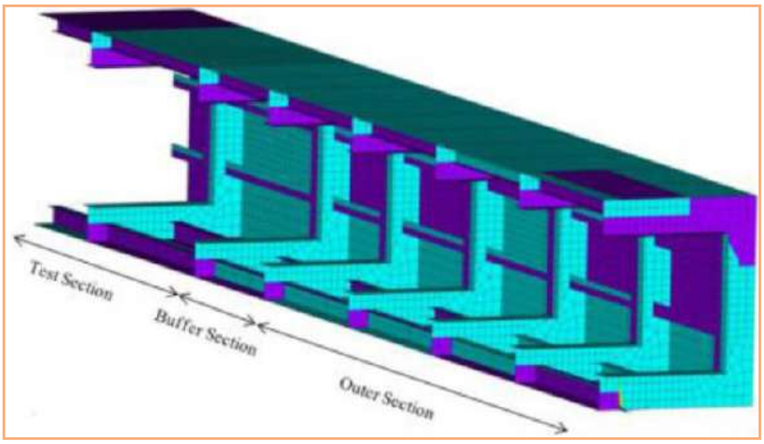


Figure 13: Element modeling applied in the case

Total elements I used in my model were 14,065 that I show their distribution in Table 4.

Item	Quantity	No. of Elements/Quantity	Total No of Elements
Test section bottom plating	I	1904	1904
Test section top plating	1	1904	1904
Buffer section bottom plating	1	68	68
Buffer section top plating	1	68	68
Outer section bottom plating	1	272	272
Outer section top plating	1	272	272
Side plating	2	1386	2772
Transverse frame's web	7	407	2849
Transverse frame's horizontal flanges	14	60	840
Transverse frame's vertical flanges	14	34	476
Longitudinal stiffener's web	10	132	1320
Longitudinal stiffener's flange	10	132	1320
Sum			14065

Table 4: Model specifications

I used a nonlinear analysis to get the ultimate strength. I used the Arc Length method in this analysis. The purpose of this analysis was to find the bending moment versus the vertical displacement of the top middle node of the test section. The specified node position and the Arc Length method are shown in Figure 14, and as it appears in the figure when the node's vertical displacement reached the 55 mm, the analysis stopped.

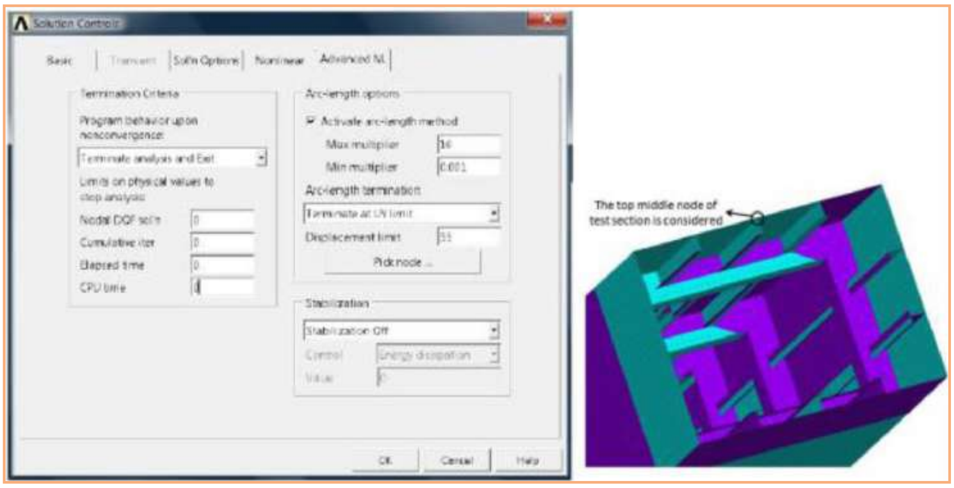


Figure 14: Arc Length method and the node positions

To explain the boundary and loading condition of my model, a three-dimensional model of Akhra's is shown in Figure 15.

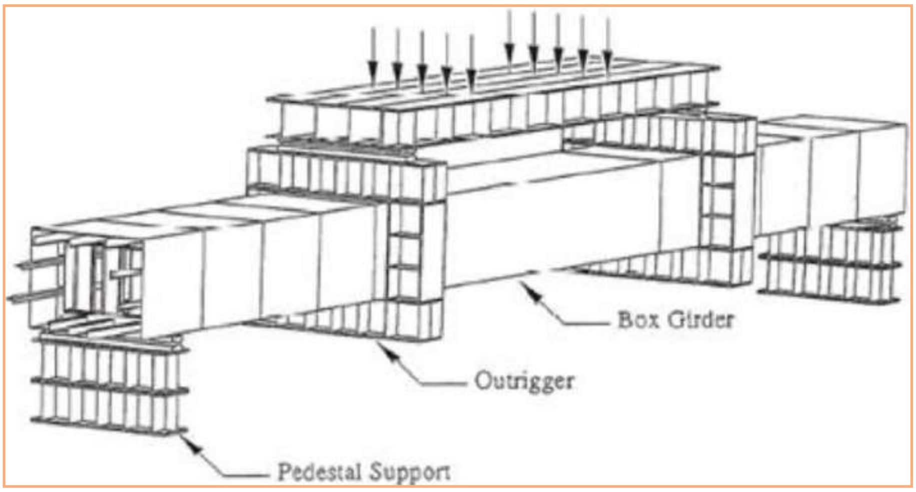


Figure 15: Akhra's three-dimensional modeling

Also, the required condition to nonlinear analysis of the finite element model, needed loading and boundary condition are presented in Figure 16.

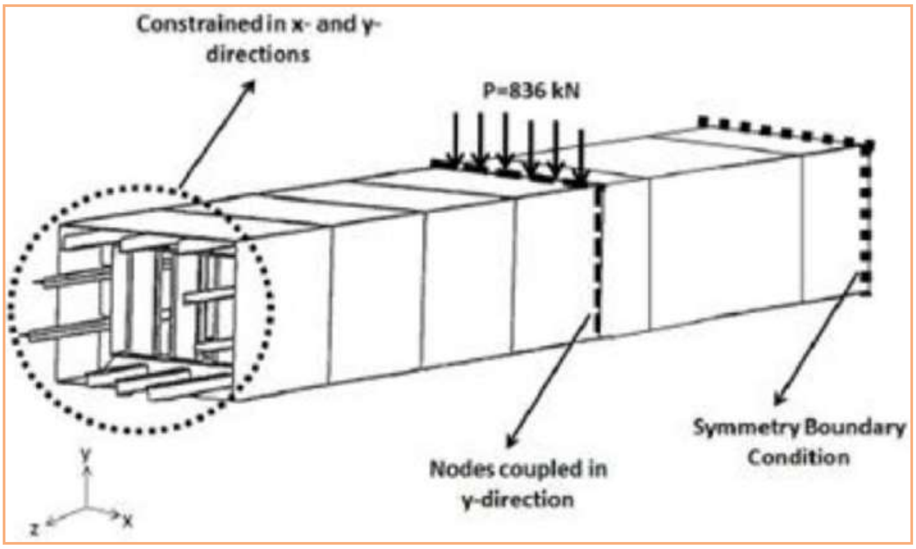


Figure 16: The required condition to nonlinear analysis of the finite element model

Regarding the simulation of the box girder's left half, the right side has a symmetry boundary condition. My model's left side is constrained along the width and height (x and y directions). In the box girder, there is a base placed under the site of loading. Thus, the loading section nodes are coupled in the vertical direction (y-direction) to prevent them from moving in this direction. Moreover, regarding this fact that Akhras's model achieved its ultimate strength at 836 kN loading, I applied this loading to my model too. Loading and boundary condition of the model are shown in Figure 17.

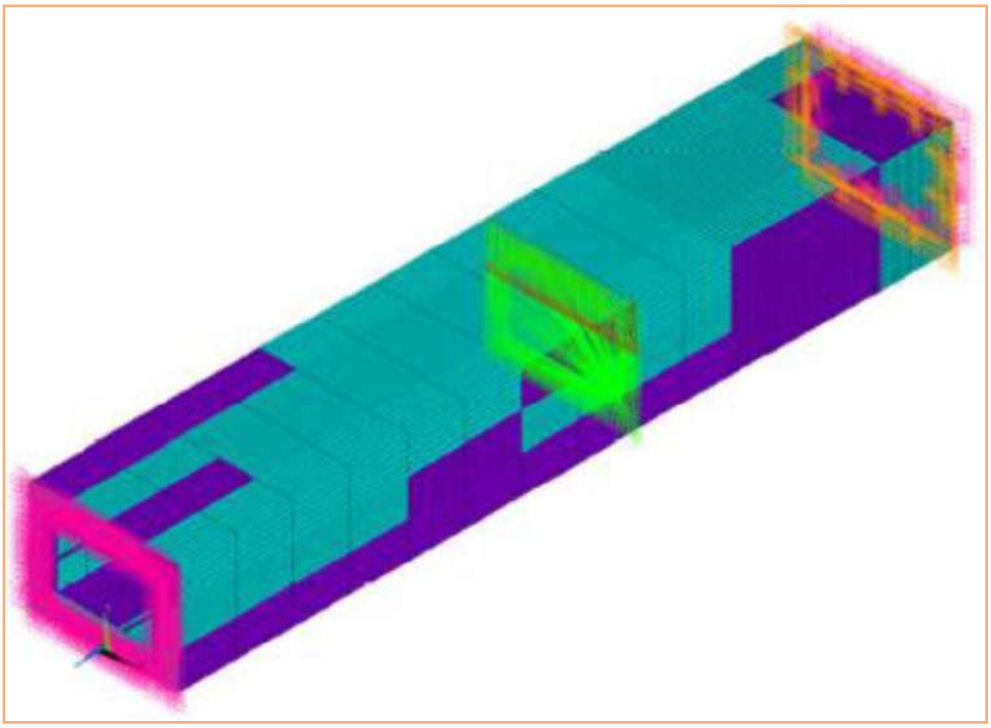


Figure 17: Loading and boundary condition of the model

I did the nonlinear analysis and gained moment variations versus displacement graph on the top middle node of the test section (as explained above), and it is shown in Figure 18. Regarding this graph, the maximum amount of bending moment is 1259.55 kN.m.

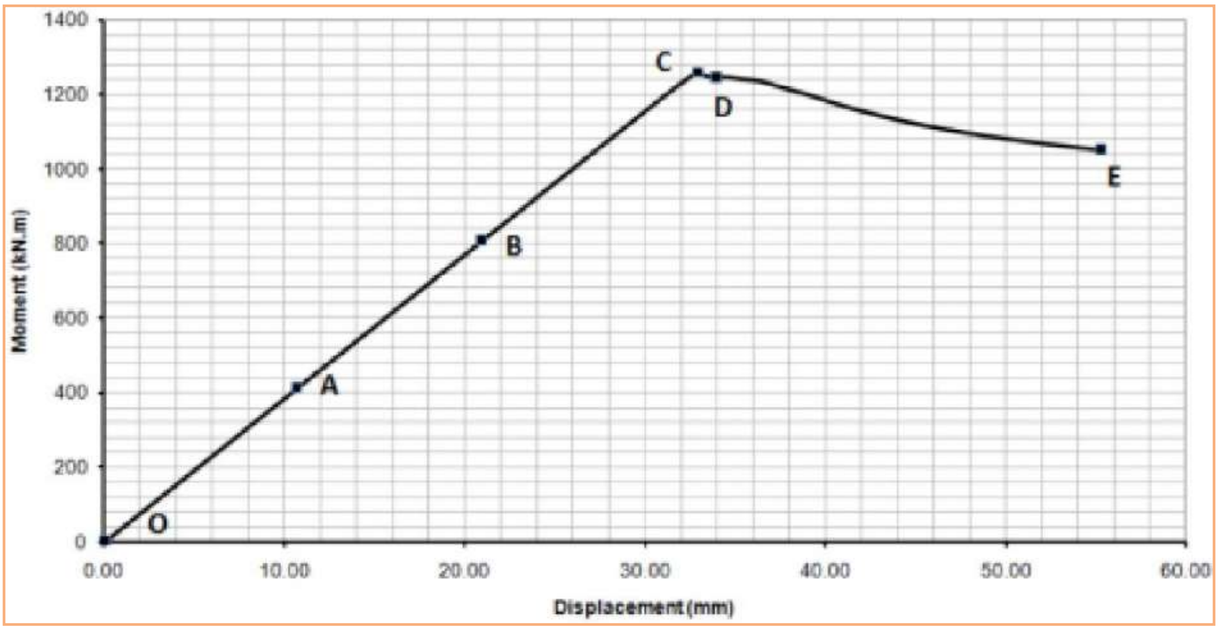


Figure 18: The moment variations versus displacement graph

I calculated the Von Mises stress distribution in the mentioned points of the Figure 18. The Von Mises stress variations, and point bending moment amounts that I calculated are presented in Figure 19.

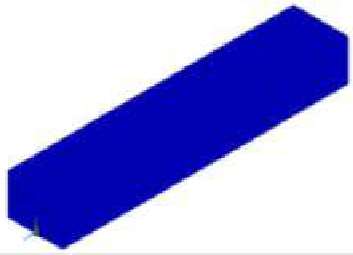
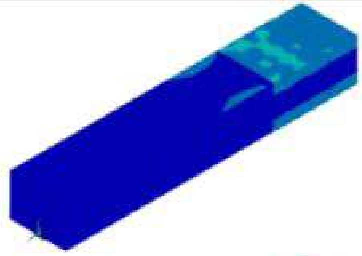
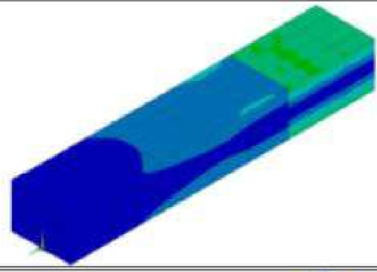
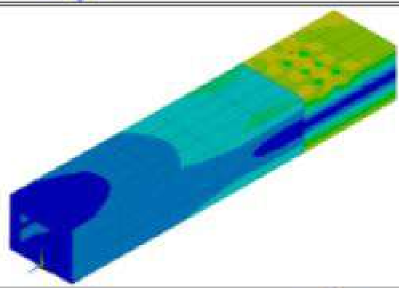
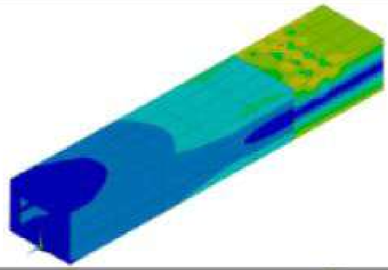
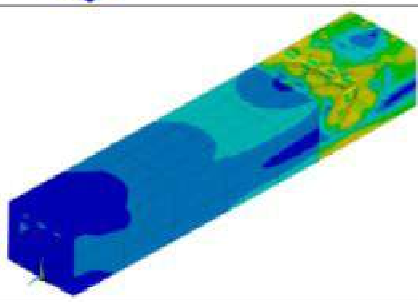
Point	Moment (kN.m)	Equivalent von Mises Stress Contours Superimposed on the Deflection Mode
O	2.47	
A	412.09	
B	807.87	
C	1259.56	
D	1249.43	
E	1051.47	

Figure 19: The Von Mises stress variations

The calculated Von Mises stress distribution in the deck, bottom, and side of the model in C point is shown in Figure 20.

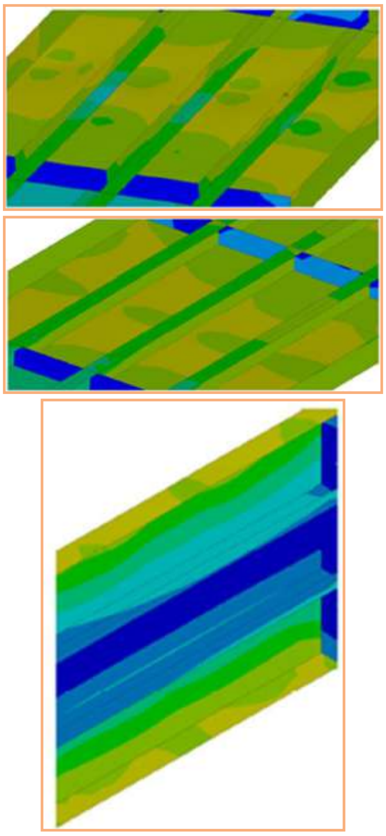


Figure 20: The calculated Von Mises stress distribution

Similarly, this stress distribution in the deck, bottom, and side of the model in E point is shown in Figure 21.

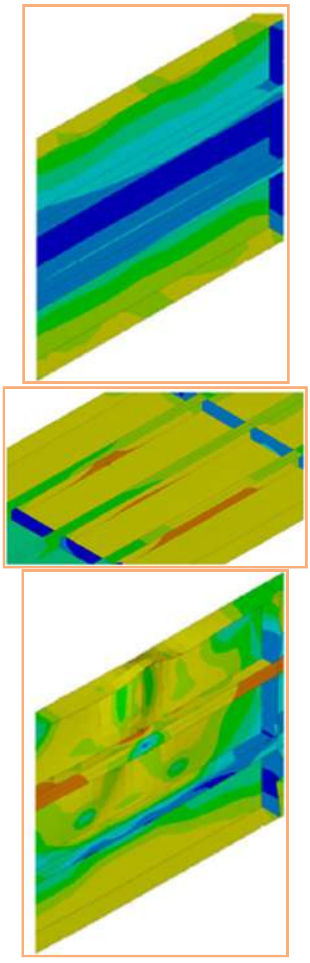


Figure 21: The stress distribution in the modeling

Deformed models regarding the Akhra's experimental test and my finite element analysis after loading are shown in Figure 22. Comparing the figures shows close conformity between the Akhra's model and the bending position of my model.

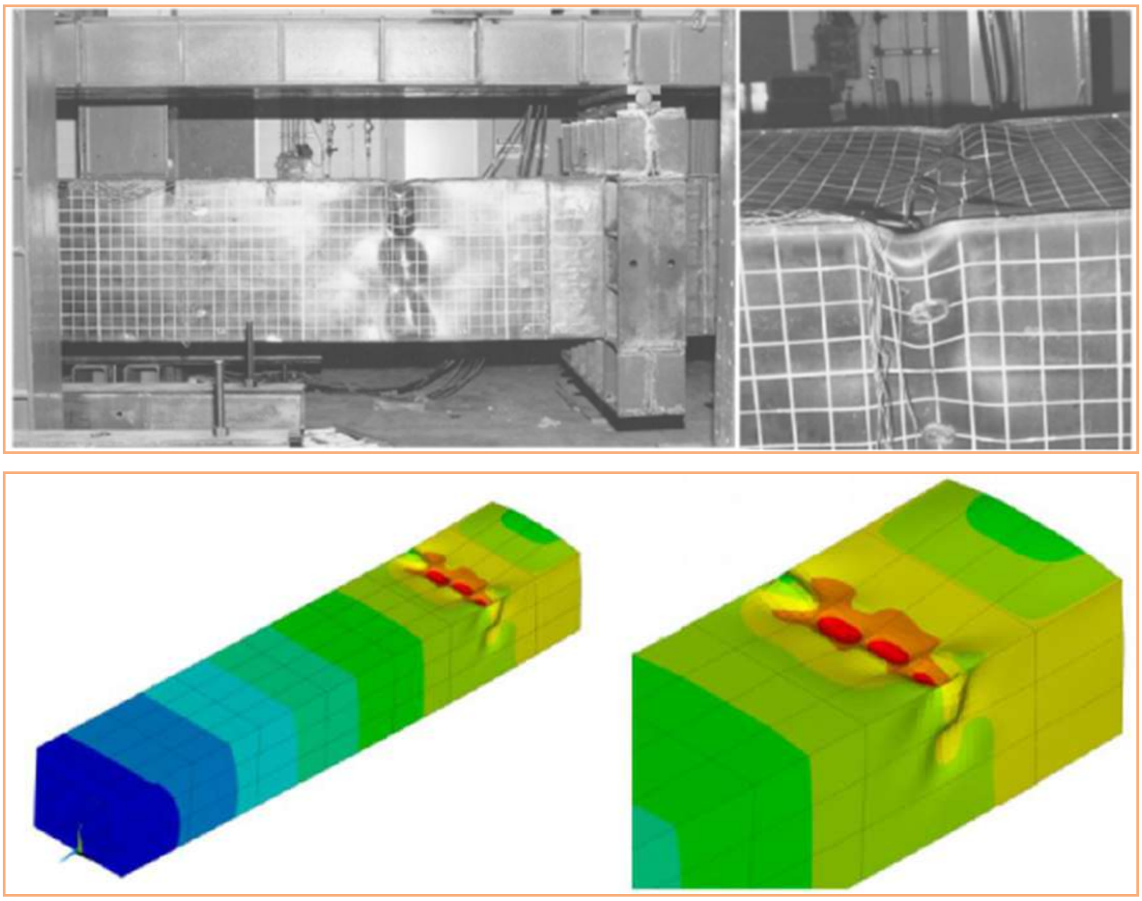


Figure 22: Deformed models regarding the Akhra's experimental test and the case study finite element analysis after loading

In Figure 23, the inner view of those two buckled models can be seen.

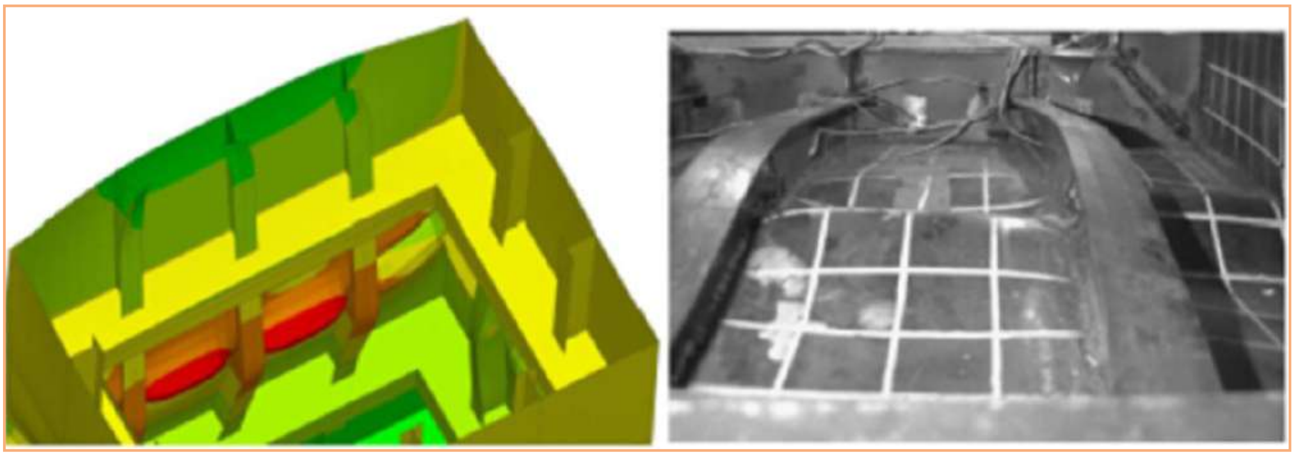


Figure 23: Inner views of the buckled modeling

The deformation process of the deck, bottom, and side of the model in E point is shown in Figure 24.

Regarding Figure 24, the interframe buckling failure in deck first happens in the simulated model under the longitudinal bending moment. Gradually increasing the load, middle cross section of test section in deck central plate, buckles leads to this extra loading transferring to longitudinal stiffeners besides this central plate. When stiffeners buckle, plates attached to them buckle too. This sequential buckling transfer reaches the deck and side junction and consequently, side plates start to buckle. This instability process happens before reaching the tensioned section of the model (bottom) to plastic mode. After this interframe buckling, general buckling happens in the model, which is clearly visible on the right side of Figure 24.

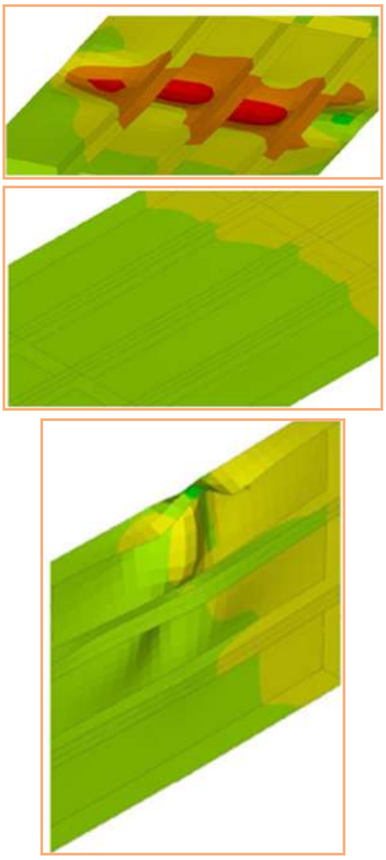


Figure 24: Deformation process in the sections of the modeling

Conclusion

As mentioned before, Akhra's reached three amounts of the ultimate moment from three methods (experimental testing, Faulkner formula, and FABSTRAN software). My results from the finite element method are compared to these three-mentioned methods in the figure below. Moreover, Qi (2005) simulated the Akhra's box girder in ANSYS, which is also shown in Figure 25.

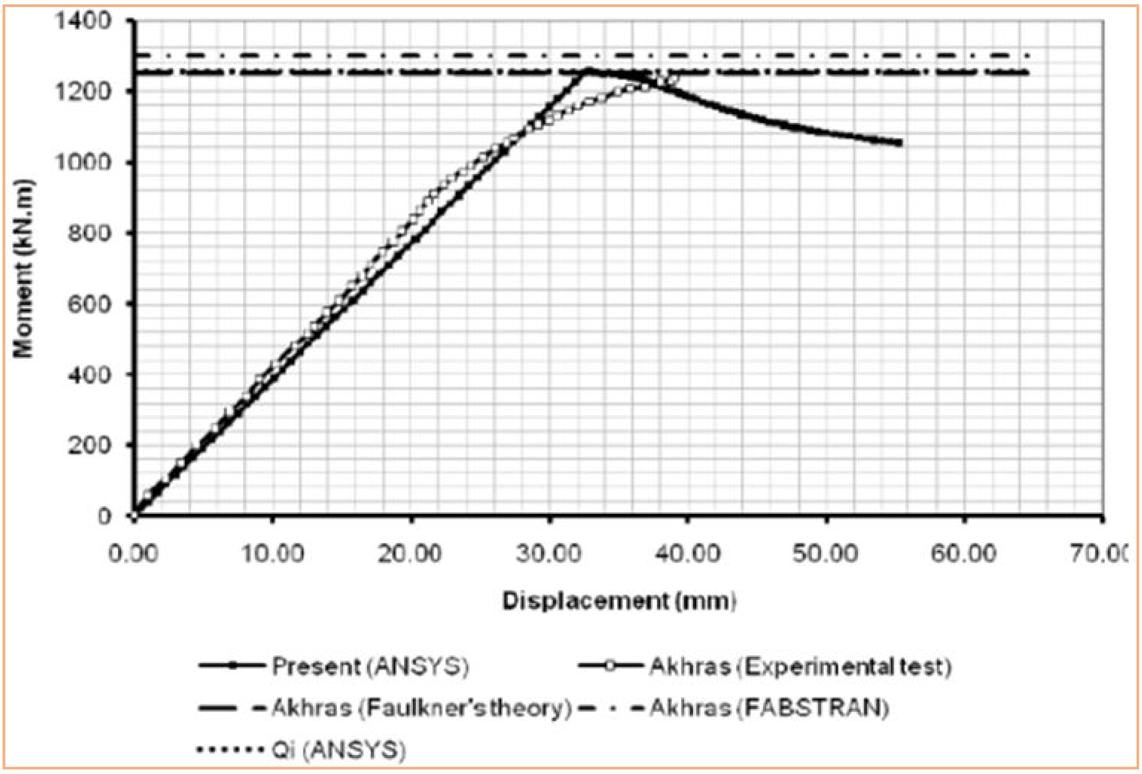


Figure 25: Akhras box girder simulation by Qi

In Table 5, I have compared my results to Akhra's and Qi research studies, and I have specified whether to consider the initial deformation and residual stresses.

Method	Ultimate Moment (KN.m)	Initial Residual Stresses	Initial Geometric Imperfections	Error with respect to Akhras's Experimental Test (%)
Akhras (Experimental test)	1238	✓	✓	-------
Akhras (Faulkner's theory)	1250	X	X	0.97
Akhras (FABSTRAN)	1300	✓	✓	5.01
Qi (ANSYS)	1256	X	✓	1.45
Present (ANSYS)	1259.55	X	✓	1.74

Table 5: The modeling result comparison to Qi and Akhra's

Regarding this table, Akhra's experimental testing on the real model and under the applied loading of 836 kN gave the 1238-kN.m result. The attained ultimate moment of the Faulkner formula, without considering the initial deformations and residual stresses was 1250 kN.m. Finally, the FABSTRAN software, which was supposed to consider initial deformations and residual stresses, gave the 1300 kN.m result. Qi considering only the initial deformations and by the ANSYS software reached the 1256 kN.m result. In my research, I also considered only initial deformations and the result was 1259.55 kN.m through the ANSYS software. Comparing my calculated moment to the real one from the Akhra's experimental model gives the 1.74 % error. Additionally, comparing the results mentioned in the above table, my research gave a better and closer result (compare to FABSTRAN software (written by Akhra's)) to the experimental testing of Akhra's. Moreover, my reached ultimate moment is very close to the one gained by Qi.

References

1. Rudolph S (2004) Theories and Applications of Plate Analysis: Classical Numerical and Engineering Methods. John Wiley & Sons.

2. Shojaee T, Salimi-Majd D, Madoliat R, Mohammadi B (2019) Development of a finite strip method for efficient prediction of buckling and post-buckling in composite laminates containing a cutout with/without stiffener Composite Structures 210: 538-52.

3. Ansys. User's Manual. Version 10.0. Houston: Swanson Analysis Systems, Inc., 2006.